CHAPTER 5

OFFHAND GRINDING OF TOOLS AND SELECTION OF CARBIDE TOOLING

CHAPTER LEARNING OBJECTIVES

Upon completing this chapter, you should be familiar with the following:

- *Identify and explain the use of grinding equipment.*
- *Identify and explain the use of grinding wheels.*
- *Identify and explain the use of single-point cutting tools.*
- *Identify and explain the use of carbide tools*

One requirement for advancement in the MR rating is the ability to grind and sharpen some of the tools used in the machine shop. The equipment used for this purpose includes bench, pedestal, carbide, and chip breaker grinders and precision grinding machines. This chapter explains the use of these grinders and how to grind small tools by using the offhand grinding technique. (We'll cover precision grinding machines in a later chapter.)

In grinding, you use the cutting action of an abrasive to remove metal. In offhand grinding you hold the workpiece in your hand and position it against the grinding surface. You must have experience and practice to do this accurately and safely. You also must know how to install grinding wheels on pedestal and bench grinders and how to sharpen or dress them.

Before you can properly grind small handtools, single-edged cutting tools, and twist drills, you must know the terms used to describe their angles and surfaces. You also must know the composition of the material from which each tool is made and the operations for which the tool is used.

Advancing technology has made carbides the dominate cutting tool in machine shops. You must understand carbide terminology and the use of carbide tools.

GRINDING SAFETY

The grinding wheel is a fragile cutting tool that operates at high speeds. Therefore, the safe operation of bench and pedestal grinders is as important as proper grinding techniques. Follow all posted safety precautions. Review your equipment operators manual for other safety precautions and any chapters of *Navy Occupational Safety and Health (NAVOSH) Program Manual for Forces Afloat,* OPNAV Instruction 5100.19B, that apply to the equipment.

BENCH AND PEDESTAL GRINDERS

Bench grinders (fig. 5-1) are small, self-contained, and usually mounted on a workbench. Use them to grind and sharpen small tools such as lathe, planer,

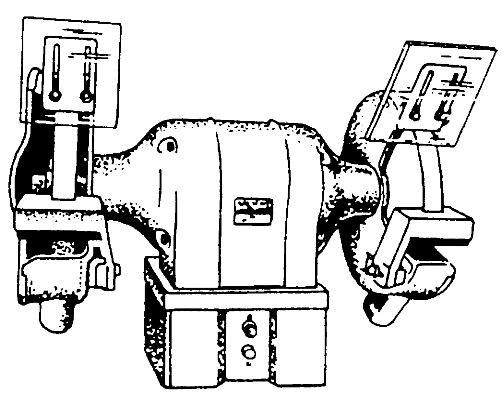


Figure 5-1.—Bench grinder.

and shaper cutting tools; twist drills; and handtools such as chisels and center punches. These grinders do not have installed coolant systems; however, a container of water is usually mounted on the front of the grinder.

Bench grinders usually have grinding wheels up to 8 inches in diameter and 1 inch thick. A wheel guard encircles the grinding wheel except for the work area. An adjustable toolrest steadies the workpiece. You can move it in or out or swivel it to adjust to grinding wheels of different diameters. An adjustable eyeshield made of safety glass should be mounted on the upper part of the wheel guard. Position this shield to deflect the grinding wheel particles away from you.

Pedestal grinders (fig. 5-2) are usually heavy-duty bench grinders mounted on a pedestal fastened to the deck. They usually have the features of a bench grinder plus a coolant system, which includes a pump, storage sump, hose, and fittings to regulate and carry the coolant to the wheel surface.

GRINDING WHEELS

A grinding wheel is made of two basic elements: (1) the abrasive grains, and (2) the bonding agent. You can think of the abrasive grains as many single-point tools embedded in a toolholder or bonding agent. Each of these grains removes a very small chip from the workpiece as it makes contact on each revolution of the grinding wheel.

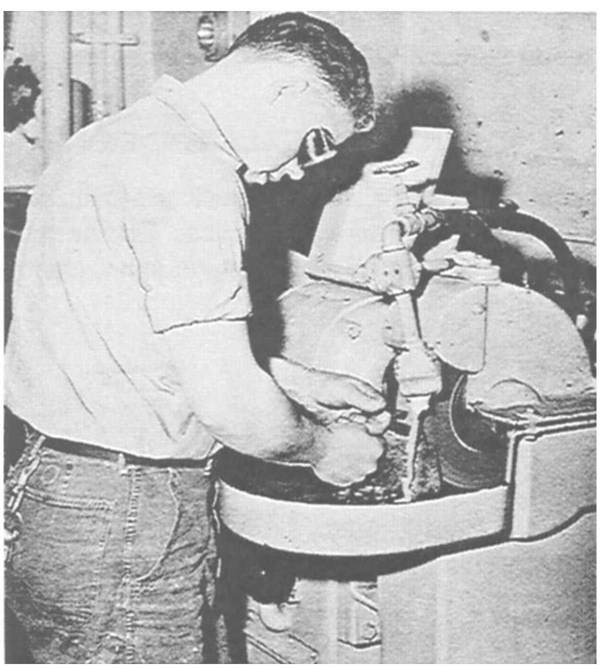
An ideal cutting tool is one that will sharpen itself when it becomes dull. This, in effect, happens to the abrasive grains. As the individual grains become dull, the pressure on them causes them to fracture and present new sharp cutting edges to the work. When the grains can fracture no more, the pressure becomes too great and they are released from the bond, allowing a new layer of sharp grains to contact the work.

SIZES AND SHAPES

The size of a grinding wheel is determined by its diameter in inches, the diameter of its spindle hole, and the width of its face. Grinding wheels have too many shapes to list in this manual, but figure 5-3 shows those used most often. The type numbers are standard and all manufacturers use them. The shapes are shown in cross-sectional views. The job will dictate the shape you should use.

WHEEL MARKINGS AND COMPOSITION

Grinding wheel markings are composed of six stations, each of which identifies a characteristic of



28.61

Figure 5-2.—Grinding on a pedestal grinder.

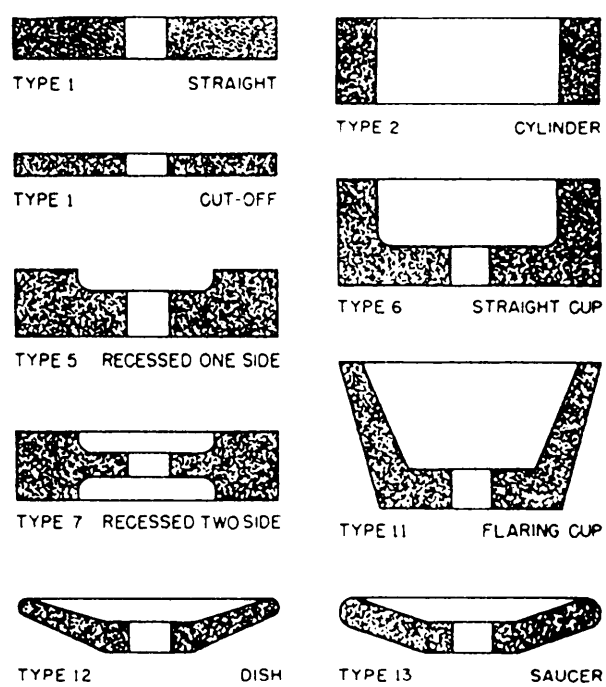


Figure 5-3.—Grinding wheel shapes.

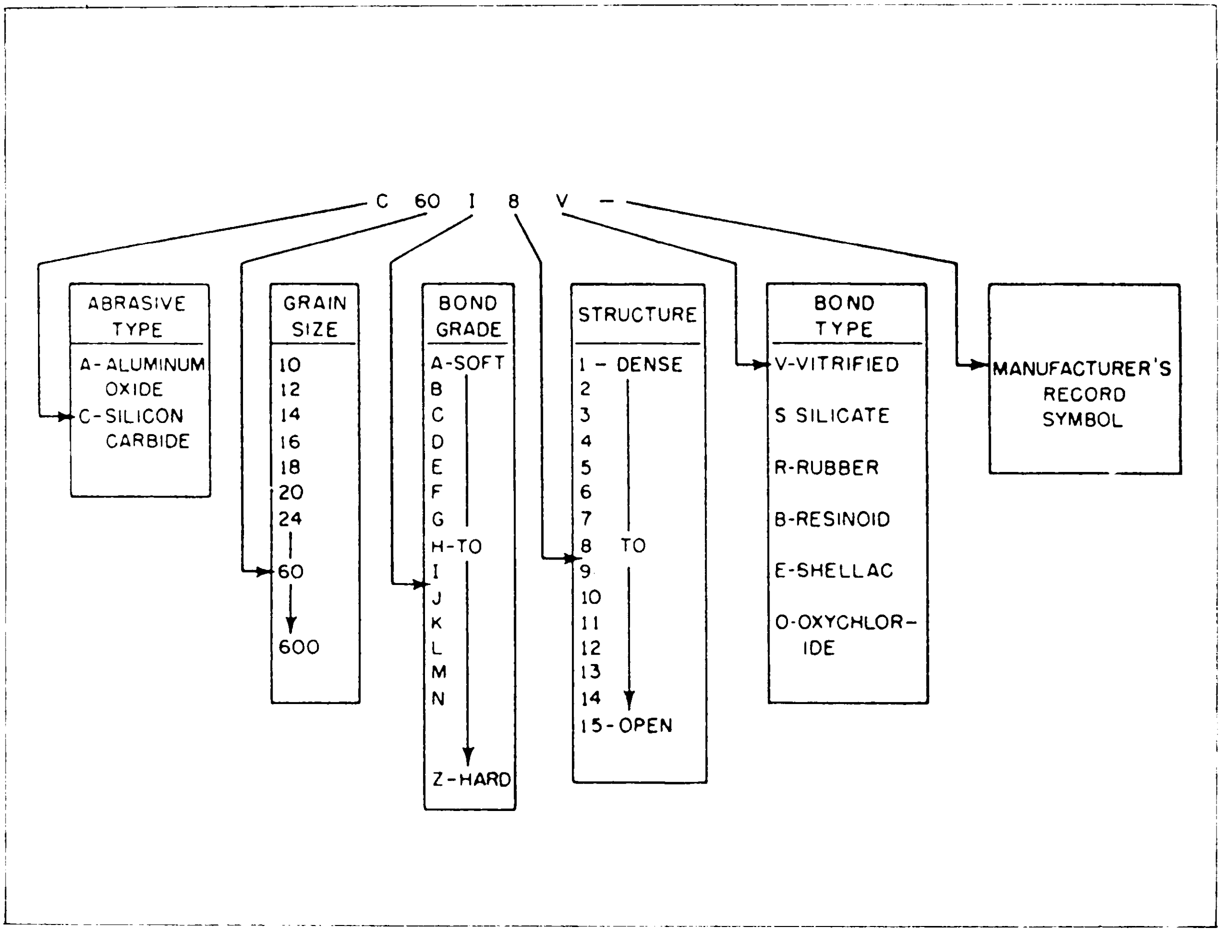


Figure 5-4.—Standard marking system for grinding wheels (except diamond).

the wheel. Since different grinding jobs require different abrasives, you should learn this identification system so you can identify the grinding wheel you need for a given job. The six stations are (1) type of abrasive, (2) grain size, (3) bond grade, (4) structure, (5) type of bond, and (6) the manufacturer's record symbol. Figure 5-4 shows the six stations that identify nearly all abrasives except diamond, which we'll explain in later paragraphs. Follow the stations in the figure from left to right as you read an explanation of each station in the following paragraphs:

1. Type of abrasive: There are two types of abrasives: natural and manufactured. Natural abrasives, such as emery, corundum, and diamond, are used only in honing stones and in special types of grinding wheels. The common manufactured abrasives are aluminum oxide and silicon carbide. They have superior qualities and are more economical than natural abrasives. Aluminum oxide (designated by the letter A in station 1) is used to grind steel and steel alloys and for heavy duty work such as to clean up steel castings. Silicon carbide (designated by the letter C in station 1) is harder but not as tough as aluminum oxide. It's used mostly to grind nonferrous metals and carbide tools. The abrasive in a grinding wheel makes up about 40 percent of the wheel.

2. Grain size: Grain sizes range from 10 to 500. The size is determined by the size of mesh of a sieve through which the grains can pass. Grain size is rated as follows: coarse: 10, 12, 14, 16, 18, 20, 24; medium: 30, 36, 46, 54, 60; fine: 70, 80, 90, 100, 120, 150, 180; and very fine: 220, 240, 280, 320, 400, 500, 600. Fine grain wheels are preferred to grind hard materials-they have more cutting edges and will cut faster than coarse grain wheels. Coarse grain wheels are generally preferred to remove metal quickly from softer materials.

3. Bond grade (hardness): The bond grade runs from A to Z, (soft to hard). It's a measure of the bond's ability to hold the abrasive grains in the wheel. A grade of soft or hard does not mean that the bond or

the abrasive is soft or hard; it means that the wheel has either a small amount of bond (soft grade) or a large amount of bond (hard grade). Figure 5-5 shows magnified portions of both soft-grade and hard-grade wheels. You can see that a part of the bond surrounds the abrasive grains, and the remainder of the bond forms into posts that hold the grains to the wheel and hold them apart from each other. The wheel with the larger amount of bonding material (hard grade) has thick bond posts and offers great resistance to grinding pressures. The wheel with the least amount of bond (soft grade) offers less resistance.

4. Structure: The structure is designated by numbers from 1 to 15. It refers to the open space between the grains, as shown in figure 5-5. Wheels with grains that are very closely spaced are said to be dense; when grains are wider apart, the wheels are said to be open. Open-grain wheels will remove more metal faster than close-grain wheels. Also, dense, or close grain, wheels normally produce a finer finish. Structure makes up about 20 percent of the grinding wheel.

5. Bond type: The bond makes up the remaining 40 percent of the grinding wheel and is one of its most important parts. The bond determines the strength of the wheel. The five basic types of bond are vitrified, silicate, rubber, resinoid, and shellac. We will describe each of them in the following paragraphs:

- Vitrified bond is designated by the letter *V.* About 75 percent of all grinding wheels are made with vitrified bond. It is not affected by oil, acid, or water. Vitrified bond wheels are strong and porous, and rapid temperature changes have little effect on them. Vitrified bond is composed of special clays. When heated to approximately 2300°F, the clays form a glasslike cement. Do NOT run vitrified bond wheels faster than 6,500 surface feet per minute (sfpm).

- Silicate bond is designated by the letter *S.* This bond is made of silicate of soda. Silicate bond wheels are used mainly on large, slow rpm machines where a cooler cutting action is wanted. Silicate bond wheels are softer than vitrified wheels, and they release the grains more readily. Silicate bond wheels are heated to approximately 500°F when they are made. Like the vitrified bond wheel, do not run this one at a speed greater than 6,500 sfpm.

- Rubber bond wheels are designated by the letter *R.* The bond consists of rubber with sulphur added as a vulcanizing agent. The bond is made into a sheet into which the grains are rolled. The wheel is stamped out of this sheet and heated in a pressurized mold until the vulcanizing action is complete. These wheels are very strong and elastic, and they are used as thin cutoff wheels. They produce a high finish and you can run them at speeds between 9,500 and 16,000 sfpm.

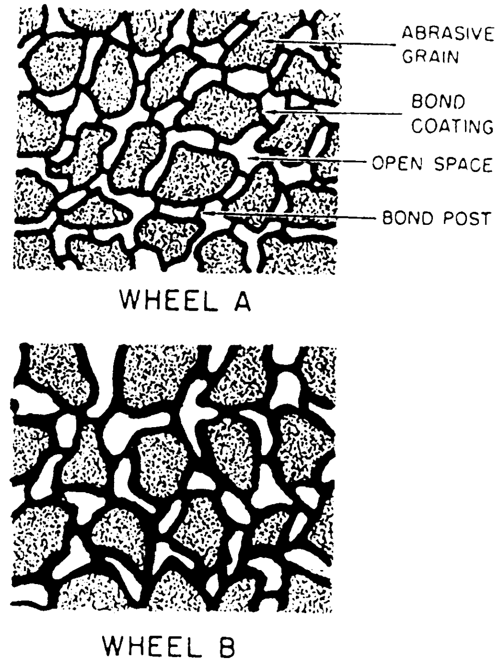


Figure 5-5.—How bond affects the grade of the wheel. Wheel A, softer; wheel B, harder.

- Resinoid bond wheels are designated by the letter *B.* Resinoid bond is made from powdered or liquid resin with a plasticizer added. The wheels are pressed and molded to size and fired at approximately 320°F. The wheels are shock resistant and very strong and they are used for rough grinding and as cutoff wheels. Like rubber bond wheels, you can run these wheels at a speed of 9,500 to 16,000 sfpm.

- Shellac bond wheels are designated by the letter *E.* They are made from a secretion from Lac bugs. The abrasive and bond are mixed, molded to shape, and baked at approximately 300°F. Shellac bond wheels give a high finish and have a cool cutting action when used as cutoff wheels. You also can run these wheels at speeds between 9,500 and 12,500 sfpm.

6. Manufacturer's Record Symbol: The sixth station of the grinding wheel marking is the manufacturer's record. This may be a letter or number, or both. The manufacturer uses it to designate bond modifications or wheel characteristics.

DIAMOND WHEELS

Diamond grinding wheels are classed by themselves. They can be made from natural or manufactured diamonds, and they are very expensive. Their cutting speeds range from 4,500 to 6,000 surface feet per minute. Use them with care and only to grind carbide cutting tools. They are marked similarly to aluminum-oxide and silicon-carbide wheels, although there is not a standard system. The usual diamond abrasive wheel identification system uses seven stations as follows:

1. Type of abrasive, designated D for natural and SD for manufactured.
2. Grit size, which can range from 24 to 500. A 100-grain size might be used for rough work, and a 220 for finish work. In a Navy machine shop, you might find a 150-grain wheel and use it for both rough and finish grinding.
3. Grade, designated by letters of the alphabet.
4. Concentration, designated by numbers. The concentration, or proportion of diamonds to bond, might be numbered 25, 50, 75, or 100, going from low to high.
5. Bond type, designated B for resinoid, M for metal, and V for vitrified.
6. Bond modification (This station may or may not be used).
7. Depth of the diamond section. This is the thickness of the abrasive layer and ranges from 1/32 to 1/4 inch. Cutting speeds range from 4,500 to 6,000 surface feet per minute.

GRINDING WHEEL SELECTION AND USE

You should select a grinding wheel that has the proper abrasive, grain, grade, and bond for the job. Base your selection on such factors as the physical properties of the material to be ground, the amount of stock to be removed (depth of cut), the wheel speed and work speed, and the finish required.

To grind carbon and alloy steel, high-speed steel, cast alloys and malleable iron, you probably should use an aluminum oxide wheel. Silicon carbide is the most suitable for nonferrous metals, nonmetallic materials, and cemented carbides.

Generally, you'll choose coarser grain wheels to grind softer and more ductile materials. Also use coarse-grain wheels to remove a large amount of material (except on very hard materials). If you need a good finish, use a fine grain wheel. If the machine you are using is worn, you may need to use a harder grade wheel to offset the effects of that wear. You also can use harder grade wheel if you use a coolant with it. Refer to your machine's operators manual to select grinding wheels for various operations.

Figure 5-6 shows the type of grinding wheel used on bench and pedestal grinders. When you replace the wheel, be sure the physical dimensions of the new wheel are correct for the grinder. Check the outside diameter, the thickness, and the spindle hole. If necessary, use an adapter (bushing) to decrease the size of the spindle hole so it fits your grinder.

You should use A3605V (coarse) and A60M5V (fine or finish) wheels to grind or sharpen single point tool bits such as those for a lathe, planer, or shaper made from high-carbon or high-speed steel. Use an A46N5V wheel for stellite tools. These wheels have aluminum oxide as an abrasive material; use them to grind steel and steel alloys only. If you use them on cast iron, nonferrous metal, or nonmetallic materials, you may load or pin the wheel when particles of the material are imbedded in the wheel's pores. This strains the wheel and could cause it to fail and possibly injure someone.

WHEEL INSTALLATION

You must install the wheel of a bench or pedestal grinder properly or it will not operate properly and may cause accidents. Before you install a wheel, inspect it for visible defects and "sound" it to learn if it has invisible cracks.

To sound a wheel, hold it up by placing a hammer handle or a short piece of cord through the spindle hole. Use a nonmetallic object such as a screwdriver handle or small wooden mallet to tap the wheel lightly on its side. Rotate the wheel 1/4 of a turn (90°) and repeat the test. A good wheel will give out a clear ringing sound. If you hear a dull thud, the wheel is cracked and should not be used.

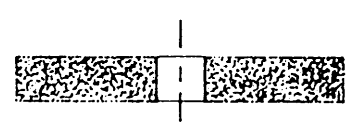


Figure 5-6.—Grinding wheel for bench and pedestal grinders.

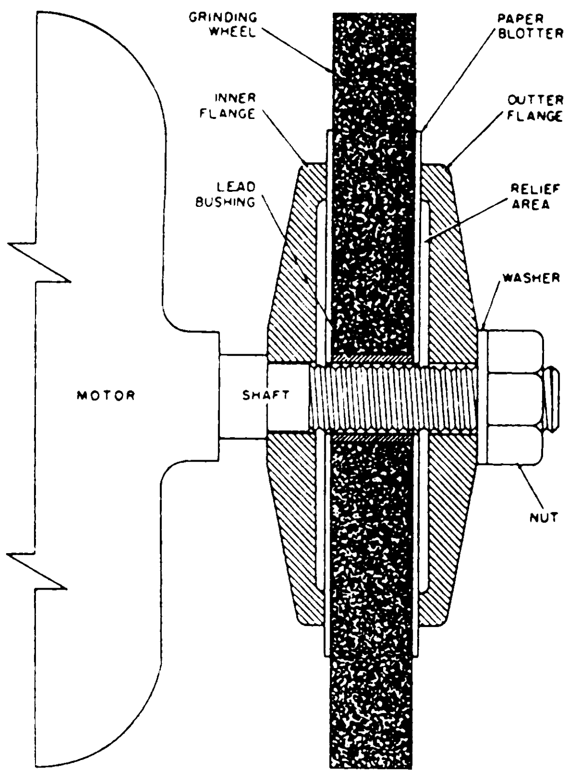


Figure 5-7.—Method of mounting a grinding wheel.

Look at figure 5-7 as you read the following explanation about wheel installation. Note that blotters are used on both sides of a wheel. A blotter ensures even pressure on the wheel and dampens the vibration between the wheel and the shaft. A paper blotter may be no more than 0.0025 inch thick and a leather or rubber blotter no more than 0.125 inch.

1. Be sure the shaft and flanges are clean and free of grit and old blotter material. Place the inner flange in place and follow it with a blotter.

2. Mount the wheel against the inner blotter and be sure it fits on the shaft without play. There should be a 0.002- to 0.005-inch clearance. You may need to scrape or ream the lead bushing in the center of the wheel to get this clearance. **NEVER FORCE THE WHEEL ONTO THE SHAFT.** You may force the wheel out of axial alignment or cause it to crack when it is used.

3. Install the second blotter, followed by the outer flange. Note that the flanges are recessed so they provide an even pressure on the wheel. The flanges should be at least one-third the diameter of the wheel.

4. Install the washer and secure the nut. Tighten the nut enough to hold the wheel firmly; if you tighten it too much, you may damage the wheel.

TRUING AND DRESSING THE WHEEL

Grinding wheels, like other cutting tools, require frequent reconditioning of cutting surfaces to perform efficiently. Dressing is the process of cleaning their cutting faces. This cleaning breaks away dull abrasive grains and smooths the surface so that there are no grooves. Truing is the removal of material from the cutting face of the wheel so that the surface runs absolutely true to some other surface such as the grinding wheel shaft.

Use the wheel dresser shown in figure 5-8 to dress grinding wheels on bench and pedestal grinders. To dress a wheel with this tool, start the grinder and let it come up to speed. Set the wheel dresser on the rest as shown in figure 5-8 and bring it in firm contact with the wheel. Move the wheel dresser across the periphery of the wheel until the surface is clean and approximately square with the sides of the wheel.

Several things can get a grinding wheel out of balance. For instance, it may be out of round, and you can usually correct the problem by dressing the wheel. Or, it may get out of balance if part of the wheel is immersed in coolant. If this happens, remove the wheel and bake it dry. If the wheel gets out of balance axially, it probably will not affect the efficiency of the wheel on bench and pedestal grinders. To correct axial unbalance, remove the wheel and clean the shaft spindle, the hole, and the flanges.

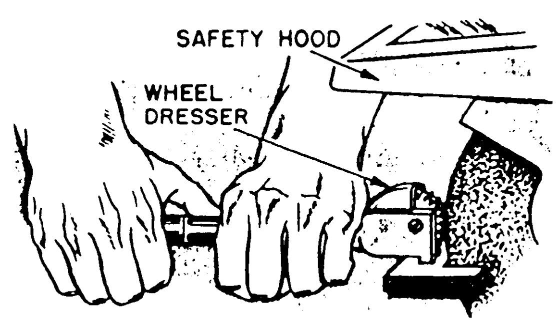


Figure 5-8.—Using a grinding wheel dresser.

WHEEL CARE AND STORAGE

It's easy to damage or break grinding wheels if you mishandle them or store them improperly. Whenever you handle them, take care not to drop them or bump them against other hard objects.

Store grinding wheels in a cabinet or on shelves large enough to allow selection of a wheel without disturbing the other wheels. The storage space should protect against high humidity, contact with liquids, freezing temperatures, and extreme temperature changes. Also secure grinding wheels aboard ship to prevent them from being damaged when the ship is at sea. Stack thin cutoff wheels on a rigid surface without any separators or blotters between them. Stack flaring cup wheels flat with the small ends together. Store all other types of wheels upright on their rims with blotters between them. A sheet metal cabinet lined with felt or corrugated cardboard to prevent wheel chipping makes good storage.

CARBIDE TOOL GRINDER

The carbide tool grinder (fig. 5-9) looks much like a pedestal grinder with the toolrest on the side instead of the front. The main components of the carbide tool grinder are a motor with the shaft extended at each end to mount the grinding wheels, the pedestal that supports the motor and is fastened to the deck, wheel guards mounted around the circumference and back of the grinding wheels as a safety device, and an adjustable toolrest mounted in front of each wheel to support the tool bits while they are being ground.

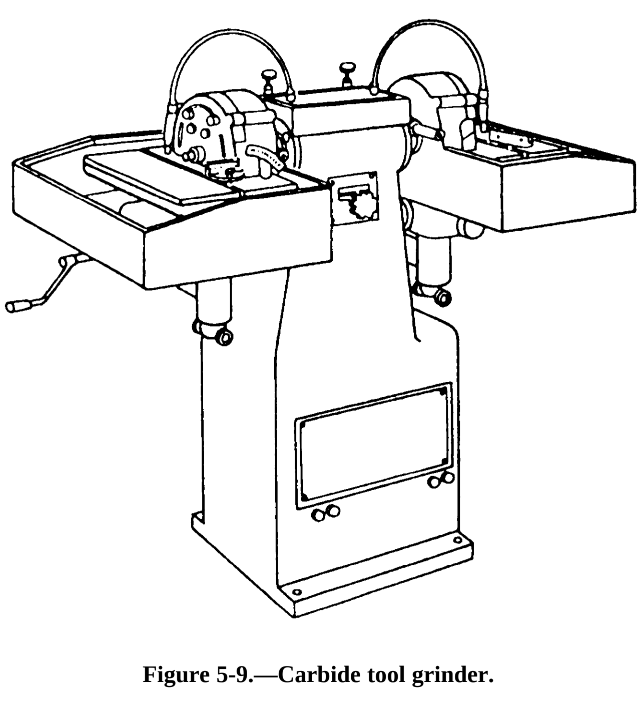


Figure 5-9.—Carbide tool grinder.

While you grind on the periphery of the wheel on a pedestal grinder, you will grind on the side of the wheel on a carbide tool bit grinder. The straight cup wheel (fig. 5-10) is similar to the wheels used on most carbide tool bit grinders. Some carbide tool grinders have a straight cup wheel on one side of the grinder and a straight wheel, such as the type used on a pedestal or bench grinder, on the other side.

The adjustable toolrest has an accurately ground groove or keyway across the top of its table. This groove holds a protractor attachment that you can set to the desired cutting edge angle. The toolrest will also adjust to permit grinding the relief angle.

If your carbide tool grinder has a coolant system, be sure you direct an ample, steady stream of coolant at the point where the tool meets the wheel. An irregular flow may allow the tool to heat up and then be quenched quickly. which may crack the carbide. If your grinder has no coolant system, let the carbide cool in the air; do **NOT** dip it in water when it becomes hot.

Carbide-tipped tool bits may have either disposable or brazed cutting edges. The disposable-tip tool bit needs no sharpening; just dispose of the tips when their cutting edges become dull. Sharpen the brazed-tip tool bit on the carbide tool bit grinder.

For best results with carbide-tipped tool bits, use a silicon-carbide wheel for roughing and a diamond impregnated wheel for finishing.

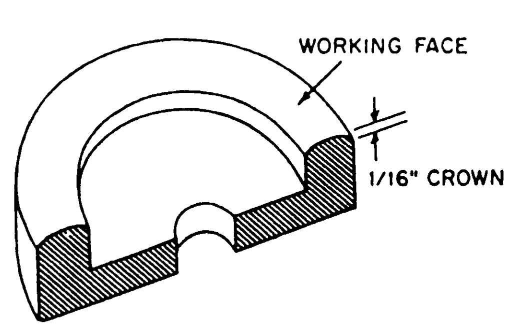


Figure 5-10.—Crown on the working face of a wheel for a carbide tool bit grinder.

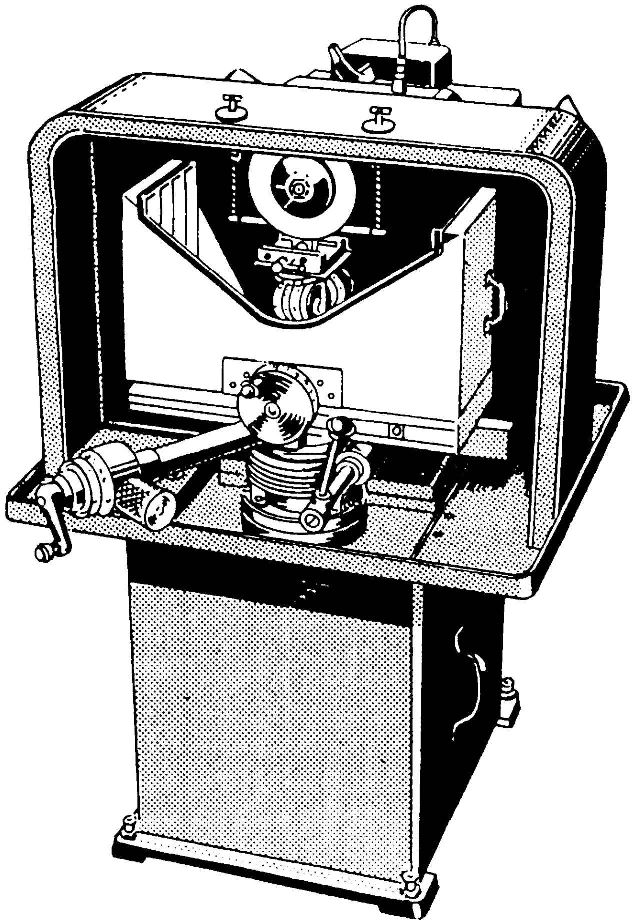


Figure 5-11.—Chip breaker grinder.

CHIPBREAKER GRINDER

A chip breaker grinder (fig. 5-11) is a specialized grinding machine. It grinds grooves or indentations on the top surface of carbide tools to control the direction and length of the chips produced in cutting metal. Later in this chapter, we'll describe the types of chip breakers that are commonly ground on carbide tools.

The chip breaker grinder has a vise you can adjust to four different angles to hold the tool to be ground. These angles are the side cutting edge, back rake, side rake, and chip breaker, and we'll explain them later in this chapter. The vise is mounted so you can move it back and forth under the grinding wheel. Both the cross feed used to position the tool under the grinding wheel and the vertical feed used to control the depth of the chip breaker are graduated in increments of 0.001 inch.

The chip breaker grinder uses a diamond wheel. It is usually a type 1 straight wheel but differs from other type 1 wheels because it is normally less than 1/4 inch thick. An SD150R100B grinding wheel is normally recommended.

Chip breaker grinders have a coolant system that either floods or slowly drips coolant onto the tool being ground. The main objective is to prevent the grinding wheel from loading up or glazing over from the grinding operation.

CUTTING TOOL MATERIALS

The materials used to make machine cutting tools must be hard enough to cut other metals, be wear resistant, have impact strength to resist fracture, and keep their hardness and cutting edge at high temperatures. Several different materials are used for cutting tools, and each one has properties different from the others. Selection of a cutting tool material depends upon the metal to be cut and the conditions under which it will be cut.

HIGH-SPEED STEEL

High-speed steel is a widely used cutting tool material. High-speed steel tools can maintain their hardness and abrasion resistance under the high temperatures and pressures generated during the general cutting process. Although the hardness of a high-speed tool (Rc 60-70) is not much greater than that of carbon-steel tools, high-speed steel begins to lose its hardness at a tempering temperature of 1,000° to 1,100°F. Machine shops generally use two types of high-speed tools: tungsten high-speed steel and molybdenum high-speed steel. These designations show the major alloying element in each of the two types. Both types resist abrasive wear, remain hard at high temperatures, and keep a similar degree of hardness. The molybdenum high-speed steel is tougher than the tungsten and is more effective in machinery operations where interrupted cuts are made. In an interrupted cut, such as cutting out-of-round or slotted material, the cutter contacts the material many times in a short period of time. This "hammering" effect dulls or breaks cutters that are not tough enough to withstand the shock effect.

CAST ALLOYS

Cast alloy tool steel usually contains varying amounts of cobalt, chrome, tungsten, and molybdenum. Tools made from these steels are generally more efficient than tools made from high-speed steel, retaining their hardness up to an operating temperature of approximately l,400°F. This characteristic allows cutting speeds approximately 60 percent greater than for high-speed steel tools. However, cast ahoy tools are not as tough as the high-speed steel tools and they cannot bear the same cutting stresses, such as interrupted cuts. Clearances ground on cast alloy cutting tools are less than those ground on high-speed steel tools because of the lower degree of toughness. Tools made from this metal are generally known as Stellite, Rexalloy, and Tantung.

CEMENTED CARBIDE

A carbide, generally, is a chemical compound of carbon and metal. The term commonly refers to cemented carbides, the cutting tools made of tungsten carbide, titanium carbide, or tantalum carbide, and cobalt in various combinations. A typical composition of cemented carbide is 85 to 95 percent carbides of tungsten and the remainder a cobalt binder for the tungsten carbide powder.

Cemented carbides are made by compressing various metal powders and sintering (heating to weld particles together without melting them) the briquettes. Cobalt powder is used as a binder for the carbide powder.

Carbides have greater hardness at both high and low temperatures than high-speed or cast alloys. At temperatures of 1,400°F and higher, carbides maintain the hardness required for efficient machining. This makes possible machining speeds of approximately 400 fpm in steels. The addition of tantalum increases the red hardness of a tool material. Cemented carbides are extremely hard tool materials (above Rc90), have a high compressive strength, and resist wear and rupture.

Cemented carbides are the most widely used tool material in the machining industry. They are particularly useful for cutting tough alloy steels that quickly break down high-speed tool steels. Various carbide grades and insert shapes are available and you should make the correct selection to machine a particular material. We'll now briefly discuss brazed-on tip carbides. Since mechanically-held tip (insert type) carbides are more widely used, we will discuss them in more depth later in the chapter.

Brazed-on Tip

The brazed-on carbide-tip cutting tool was the first carbide cutting tool developed and made available to the metal cutting industry. A brazed-on tip can be easily ground to machine such jobs. The various styles of tools required in machinery, such as turning, facing, threading, and grooving are available with different grades of carbide tips already brazed onto steel shanks. You can also get small carbide blanks and have them brazed onto shanks.

When you use cutting tools with brazed-on carbide tips, chip control may be provided by either feeds and speeds or by chip breaker grooves ground into the top of the carbide tip. The best way to grind a chip breaker is to use a chip breaker grinder with a diamond impregnated wheel. The depth of the chip breakers averages about 1/32 inch, while the width varies with the feed rate, depth of cut and material being cut. Grind the chip breaker narrow at first and widen it if the chip does not curl and break quickly enough. You may also use these same types of chip breakers on high-speed steel cutters.

Mechanically Held Tip (Insert Type)

Mechanically held carbide inserts are available in several different shapes—round, square, triangular, diamond threading, and grooving—and in different thicknesses, sizes, and nose radii. In the following paragraphs, we'll discuss the most important criteria you'll need to select an insert.

OPERATING CONDITIONS.—You must use three variables to establish metal removal rate: speed, feed, and depth of cut. Cutting speed has the greatest effect on tool life. A 50 percent increase in cutting speed will decrease tool life by 80 percent. A 50 percent increase in feed will decrease tool life by 60 percent. The cutting edge engagement or depth of cut is limited by the size and thickness of the carbide insert and the hardness of the material being machined. Hard materials require decreased feed, speed, and depth of cut. The depth of cut is limited by the strength and thickness of the carbide insert, the rigidity of the machine and setup, the horsepower of the machine, and the amount of material to be removed.

Edge wear and cratering are the most frequent tool breakdowns and they occur when friction and abrasion break down the tool relief surface. They are also caused by the tearing away of minute carbide particles from the built-up edge. The cutting edge is usually chipped or broken in this case. Lack of rigidity, too much feed, or too slow a speed causes chipped or broken inserts.

Thermal shock is caused by sudden heating and cooling that causes a tool to crack, then break. This

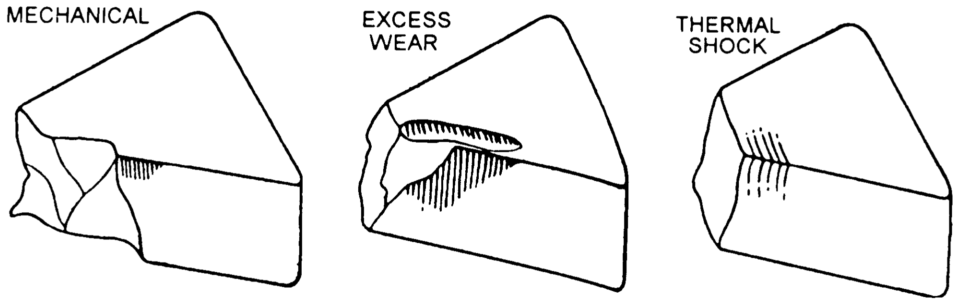


Figure 5-12.—Three causes of tool breakage.

most likely will occur when an inadequate amount of coolant is used. If you can't keep the work and tool flooded with coolant, it's usually better to machine dry.

Figure 5-12 shows three of the causes of tool breakage.

The following list shows some of the things you can do to overcome the problems of tool edge breakdown:

If edge wear occurs:

- Decrease machining speed.
- Increase feed.
- Change to a harder, more wear-resistant carbide grade.

If the cutting edge is chipped or broken:

- Increase speed.
- Decrease feed and/or depth of cut.
- Change to a tougher grade carbide insert.
- Use negative rake.
- Hone the cutting edge before use.
- Check the rigidity and tooling overhang.

When there is buildup on the cutting edge.

- Increase speed.
- Change to a positive rake tool.
- Change to a grade containing titanium.
- Increase the side cutting edge angle.
- Decrease feed.

CEMENTED CARBIDE GRADES.—Cemented carbides have been organized into grades. Properties that determine grade include hardness, toughness, and resistance to chip welding or cracking, The properties of carbide tools may vary by the percentages of cobalt and titanium or tantalum carbides. Properties may also vary during the processing by the grain size of carbides, density, and other modifications. Some tungsten carbide inserts are given a titanium carbide coating (about 0.0003 in. thick) to help them resist cratering and edge breakdown.

The grades of carbides have been organized according to their suitable uses by the *Cemented Carbide Producers Association* (CCPA). When you select a carbide, use a table made up of those suitable uses rather than make your choice based on composition. The following list shows cemented carbide grades with specific chip removal applications:

C-1 Roughing cuts (cast iron and nonferrous materials

C-2 General purpose (cast iron and nonferrous materials

C-3 Light finishing (cast iron and nonferrous materials

C-4 Precision boring (cast iron and nonferrous materials

C-5 Roughing cuts (steel)

C-6 General purpose (steel)

C-7 Finishing cuts (steel)

C-8 Precision boring (steel)

The hardest of the nonferrous/cast iron grades is C-4 and the hardest of the steel grades is C-8. This system does not specify the particular materials, alloy, or machining operations. It combines experience gained from using carbides and manufacturer's recommendations to select the proper grade of carbide.

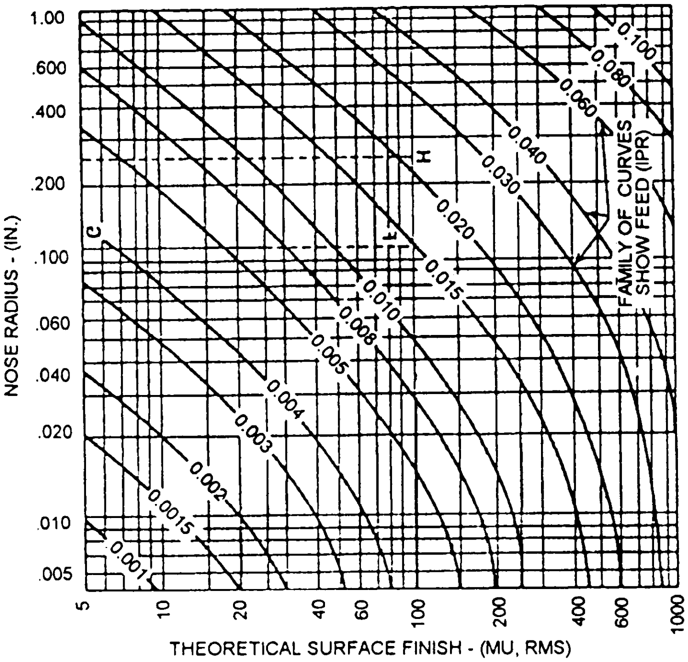


Figure 5-13.—Surface finish vs nose radius chart.

NOSE RADIUS.—Selecting the nose radius can be important because of tool strength, surface finish, or perhaps the need to form a fillet or radius on the work. To determine the nose radius according to strength requirements, use the chart shown in figure 5-13. Consider that the feed rate, depth of cut, and workpiece condition determine strength requirements.

Large radii are strongest and can produce the best finishes, but they also can cause chatter between tool and workpiece. For example, the dashed line on the chart shows that a 1/8-inch radius would be required

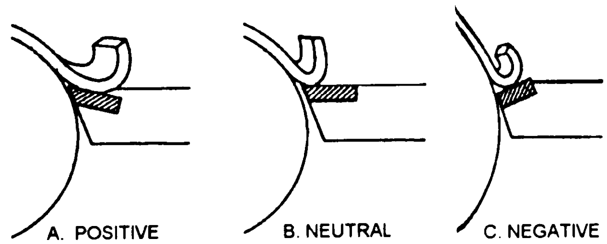


Figure 5-15.—Side view of rake angles.

for turning at a feed rate of 0.015 inch to obtain a 100 microinch finish. A 1/4-inch radius would be required with a 0.020-inch feed rate.

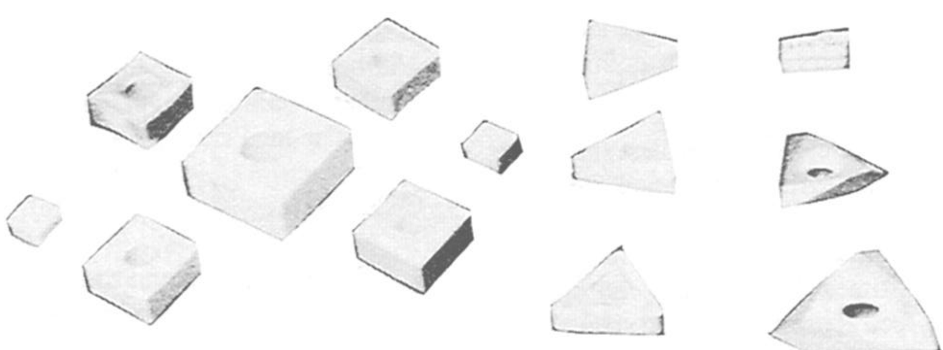
INSERT SHAPES.—Indexable inserts (fig. 5-14) are clamped in toolholders of various designs. Each of these inserts has several cutting edges. After you have used all of the edges, discard the insert.

Round inserts have the greatest strength and, like large radius inserts, they offer higher feed rates with equal finishes.

Square inserts have lower strength and fewer possible cutting edges than round tools, but they are much stronger than triangular inserts

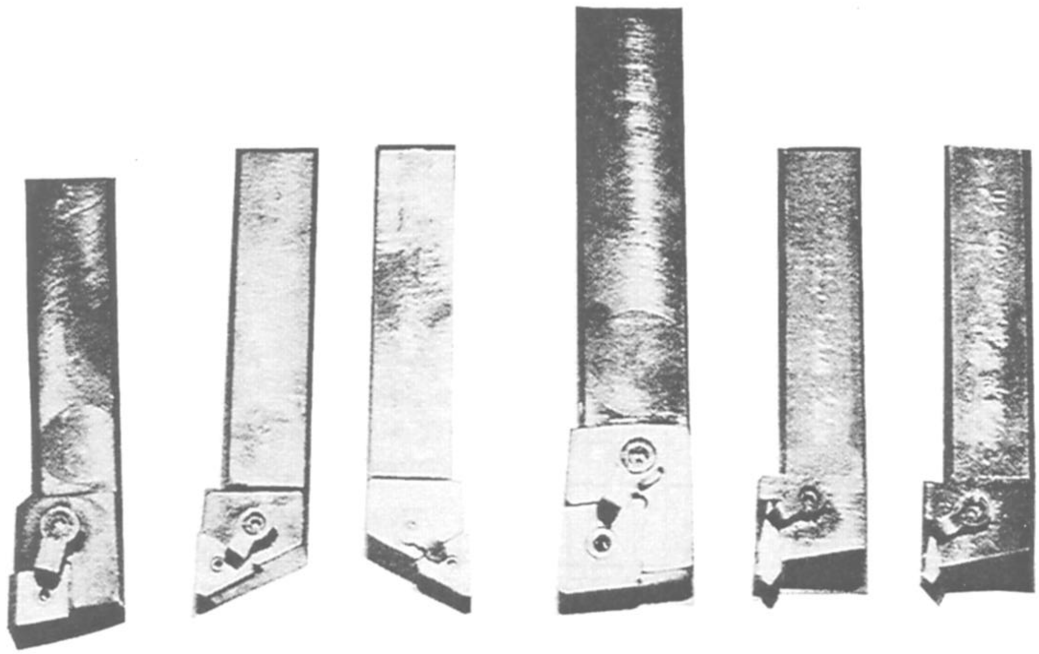
Triangular inserts have the greatest versatility. For example, you can use them in combination turning and facing operations where round or square inserts are often not adaptable to such combinations. Because the included angle between cutting edges is less than 90°, you also can use triangular inserts for tracing operations. The main disadvantages are their reduced strength and fewer cutting edges per insert.

RAKE ANGLE.—When selecting rake angles (fig. 5-15), you need to consider the machining conditions. Use negative rake where there is maximum



28.479

Figure 5-14.—Indexable inserts.



28.480

Figure 5-16.—Toolholders for carbide inserts.

rigidity of the tool and work and where you can maintain high machining speeds. Negative rake tools require more horsepower. Under these conditions, negative rake tools are stronger and produce satisfactory results.

You may use negative rake inserts on both sides, doubling the number of cutting edges per insert. This is possible because end and side relief are provided by the angle of the toolholder rather than by the shape of the insert.

Use positive rake inserts where rigidity of the tool and work is reduced and where high cutting speeds are not possible; for example, on a flexible shaft of small diameter. Positive rake tools cut with less force so deflection of the work and toolholder should be reduced. High cutting speeds (sfpm) are often not possible on small diameters because of limitations in spindle speeds.

INSERT SIZES.—Select the smallest insert that can sustain the required depth of cut and feed rate. The depth of cut should always be as great as possible. The rule of thumb is to select an insert with cutting edges 1 1/2 times the length of cutting edge engagement. The feed for roughing mild steel should be approximately 1/10 the depth of cut.

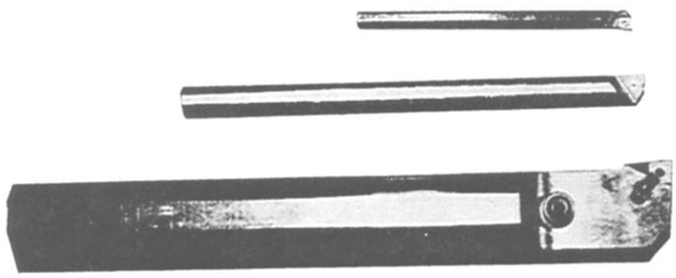
TOOLHOLDER AND BORING BAR STYLES.—Tool style pertains to the configuration of toolholders and boring bars used to hold a carbide insert. To determine style, you need to know something about the particular machine tool you are using and the operations you will do on it. Figures 5-16 and 5-17 show some of the styles available for toolholders and boring bars.

TOOLHOLDER AND CARBIDE INSERT IDENTIFICATION.—The carbide and toolholder manufacturers and the *American Standards Association* (ASA) have adopted a system to identify toolholders and inserted carbides. The system is used to call out the toolholder geometry and to identify inserts. You’ll find these charts in manufacturer’s catalogs; copy them and keep them in your toolbox.

We have given you an overview on carbides; certainly not everything you need to know. You also must work with personnel who know how to use them and then learn by using them yourself. Also, a number of carbide manufacturers offer schools to help you understand carbides and their uses.

CERAMIC

Other than diamond tools, ceramic cutting tools are the hardest and most heat resistant available to the machinist. A ceramic cutting tool can machine metals that are too hard for carbide tools, and they can sustain cutting temperatures up to 2,000°F. Therefore, you can use ceramic tools at cutting speeds two to four times greater than cemented carbide tools.



28.481

Figure 5-17.—Boring bars for carbide inserts.

Ceramic cutting tools are available as either solid ceramic or as ceramic coated carbide. They come in several of the insert shapes available in cemented carbides and they are secured in the toolholder by a cl amp.

Whenever you handle ceramic cutting tools, be very careful because they are very brittle and will not tolerate shock or vibration. Be sure your lathe setup is very rigid and do not take interrupted cuts. Also be sure the lathe feed rate does not exceed 0.015 to 0.020 inch per revolution. Any greater rate will subject, the insert to excessive forces and may cause it to fracture.

GROUND SINGLE-POINT, HIGH-SPEED CUTTING TOOLS

A single-point or single-edged cutting tool has only one cutting edge as opposed to two or more on other tools. Drill bits are multiple-edged cutters; most lathe tools are single edged. To properly grind a single-point cutting tool, you must know the relief angles, the rake angles, and the cutting edge angles that are required for specific machines and materials. You also must know what materials are generally used as cutting tools and how tools for various machines differ.

CUTTING TOOL TERMINOLOGY

Figure 5-18 shows the application of the angles and surfaces we use to discuss single-point cutting tools. Notice there are two relief angles and two rake angles and that the angle of keenness is formed by grinding a rake angle and a relief angle. We'll discuss

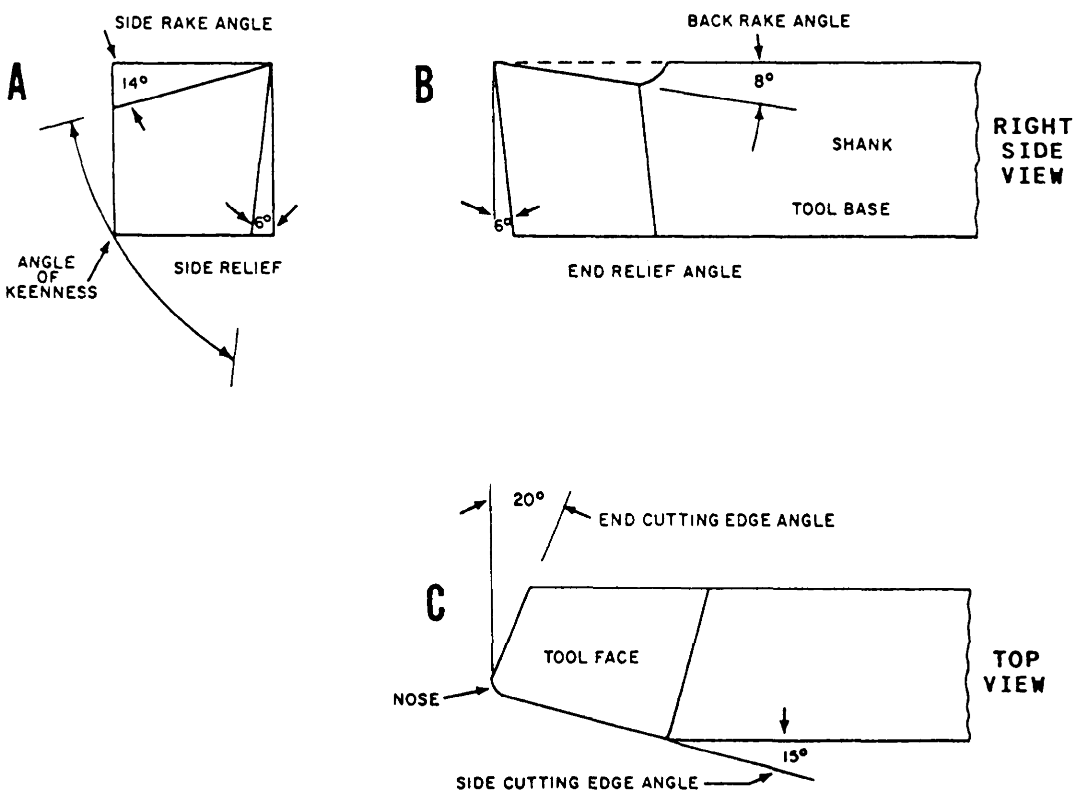


Figure 5-18.—Applications of tool terminology.

the angles that have to be ground on a high-speed single-point cutting tool in the following paragraphs.

Side Rake

Side rake is the angle at which the top surface of the tool bit is ground away making a slope either away from or toward the side cutting edge. Figure 5-18, view A, shows a positive side rake angle. When the side rake is ground toward the side cutting edge, the side rake has a negative angle. The amount of side rake influences to some extent the size of the angle of keenness. It causes the chip to “flow” to the side of the tool away from the side cutting edge. A positive side rake is most often used on ground single-point tools. Generally, the side rake angle will be steeper (in the positive direction) to cut the softer metals and will decrease as the hardness of the metal increases. A steep side rake angle in the positive direction causes the chip produced in cutting to be long and stringy. Decreasing the angle will cause the chip to curl up and break more quickly. A negative side rake is recommended when the tool will be subjected to shock, such as an interrupted cut or when the metal being cut is extremely hard.

Back Rake

The back rake is the angle at which the top surface of the tool is ground away mainly to guide the direction of the flowing chips. It is ground primarily to cause the chip to “flow” back toward the shank of the tool. Back rake may be positive or negative. It’s positive (fig. 5-18, view B) if it slopes downward from the nose toward the shank, and it’s negative if a reverse angle is ground. The rake angles help form the angle of keenness and direct the chip flow away from the point of cutting. The same general recommendations concerning positive or negative side rake angles apply to the back rake angle.

Side Relief

The side relief (fig. 5-18, view A) is the angle at which the side of the tool is ground to prevent the tool bit from rubbing into the work. The side relief angle, like the side rake angle, influences the angle of keenness. A tool with proper side relief causes the side thrust to be concentrated on the cutting edge rather than rub on the flank of the tool.

End Relief

The end relief (fig. 5-18, view B) is the angle at which the end surface of the tool is ground so that the front face edge of the tool leads the front surface.

Angle Of Keenness

The angle of keenness or wedge angle (fig. 5-18, view A) is formed by the side rake and the side relief ground in a tool. The angle of keenness is equal to 90° minus the sum of the side rake and side relief angles. Generally this angle is smaller for cutting soft materials.

Side Cutting Edge

The side cutting edge angle (fig. 5-18, view C) is ground on the side of the tool that is fed into the-work. This angle can vary from 0° for cutting to a shoulder, up to 30° for straight turning. An angle of 15° is recommended for most rough turning operations. In turning long slender shafts, a side cutting edge angle that is too large can cause chatter. Since the pressure on the cutting edge and the heat generated by the cutting action decrease as the side cutting edge angle increases, the angle should be as large as the machining operation will allow.

End Cutting Edge

The end cutting edge angle (fig. 5-18, view C) is ground on the end of the tool to permit the nose to make contact with the work without the tool dragging the surface. An angle of from 8° to 30° is commonly used, with approximately 15° recommended for rough turning operations. Finish operations can be made with the end cutting edge angle slightly larger. Too large an end cutting edge angle will reduce the support given the nose of the tool and could cause premature failure of the cutting edge.

Nose

The nose (fig. 5-18, view C) strengthens the tip of the tool, helps to carry away the heat generated by the cutting action, and helps to obtain a good finish. A tool whose nose is ground to a straight point will fail much more rapidly than one with a slight radius or rounded end ground or honed on it. However, too large a radius will cause chatter because of excessive tool contact with the work. Normally, you should use

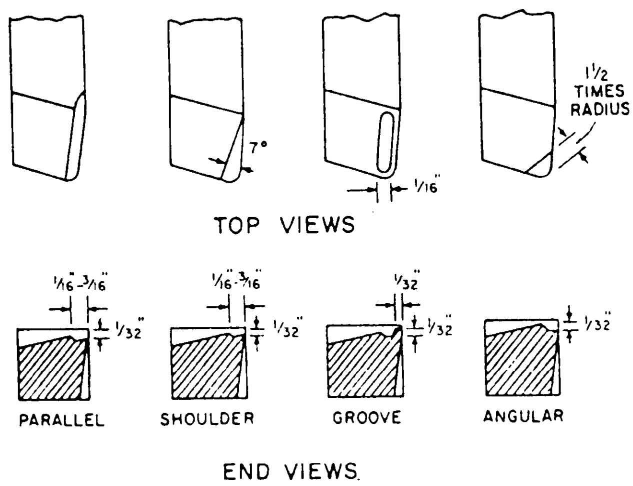


Figure 5-19.—Chip breakers.

a radius of from 1/64 to 1/32 inch in turning operations.

GROUND-IN CHIP BREAKERS

Chip breakers are indentations ground on the top surface of the tool to reduce or prevent the formation of long and dangerous chips. The chip breaker will cause the chips to curl up and break into short, safe, manageable chips. You normally grind chip breakers on roughing tools, but you can grind them on finishing tools used to machine soft ductile metals. Figure 5-19 shows four of the several types of chip breakers.

The dimensions given are general and can be modified to compensate for the various feed rates, depths of cut, and types of material being machined. Grind the groove-type chip breaker carefully to prevent it from coming too close to the cutting edge; that reduces the life of the tool because there is less support of the cutting edge. You can use the diamond wheel on a chip breaker grinder to grind carbide-tipped tools. Grind high-speed tools with an aluminum oxide grinding wheel. You can use a bench grinder for this purpose if you dress the wheel until it has a sharp edge. Or, you can clamp the tool in a universal vise that you can set to compound angles on a surface or tool and cutter grinder.

SHAPES OF HIGH-SPEED ENGINE LATHE CUTTING TOOLS

Figure 5-20 shows the most popular shapes of ground high-speed lathe tool cutter bits and their applications. Each of the types shown is described in the following paragraphs:

- **Left-hand turning tool:** Grind this tool for machining work by feeding it from left to right as shown in figure 5-20, view A. The cutting edge is on the right side of the tool and the top of the tool slopes down away from the cutting edge.

- **Round-nose turning tool:** This tool is for general all-round machine work and is used to make light roughing cuts and finishing cuts. You should usually grind the top of the cutter bit with side rake so the tool may be fed from right to left. You may sometimes grind the cutter bit flat on top so you can feed the tool in either direction (fig. 5-20, view B).

- **Right-hand turning tool:** This is just the opposite of the left-hand turning tool and is designed to cut when fed from right to left (fig. 5-20, view C). The cutting edge is on the left side. This is an ideal tool for roughing cuts and general all-round machine work.

- **Left-hand facing tool:** Use this tool for facing on the left-hand side of the work as shown in figure 5-20, view D. The direction of feed is away from the lathe center. The cutting edge is on the right-hand side of the tool and the point of the tool is sharp to permit machining a square corner.

- **Threading tool:** Grind the point of the threading tool to a 60° included angle to machine V-form screw threads (fig. 5-20, view E). Usually, you should grind the top of the tool flat and leave clearance on both sides of the tool so it will cut on both sides.

- **Right-hand facing tool:** This tool is just the opposite of the left-hand facing tool. Use it to face the right end of the work and to machine the right side of a shoulder. (See fig. 5-20, view F.)

- **Square-nosed parting (cut-off) tool:** The principal cutting edge of this tool is on the front. (See fig. 5-20, view G.) Both sides of the tool must have enough clearance to prevent binding and should be ground slightly narrower at the back than at the cutting edge. Use this tool to machine necks and grooves, square corners, and to cut off.

- **Boring tool:** Usually, you should grind a boring tool in the same shape as the left-hand turning tool so the cutting edge is on the front side of the cutter bit and may be fed in toward the headstock.

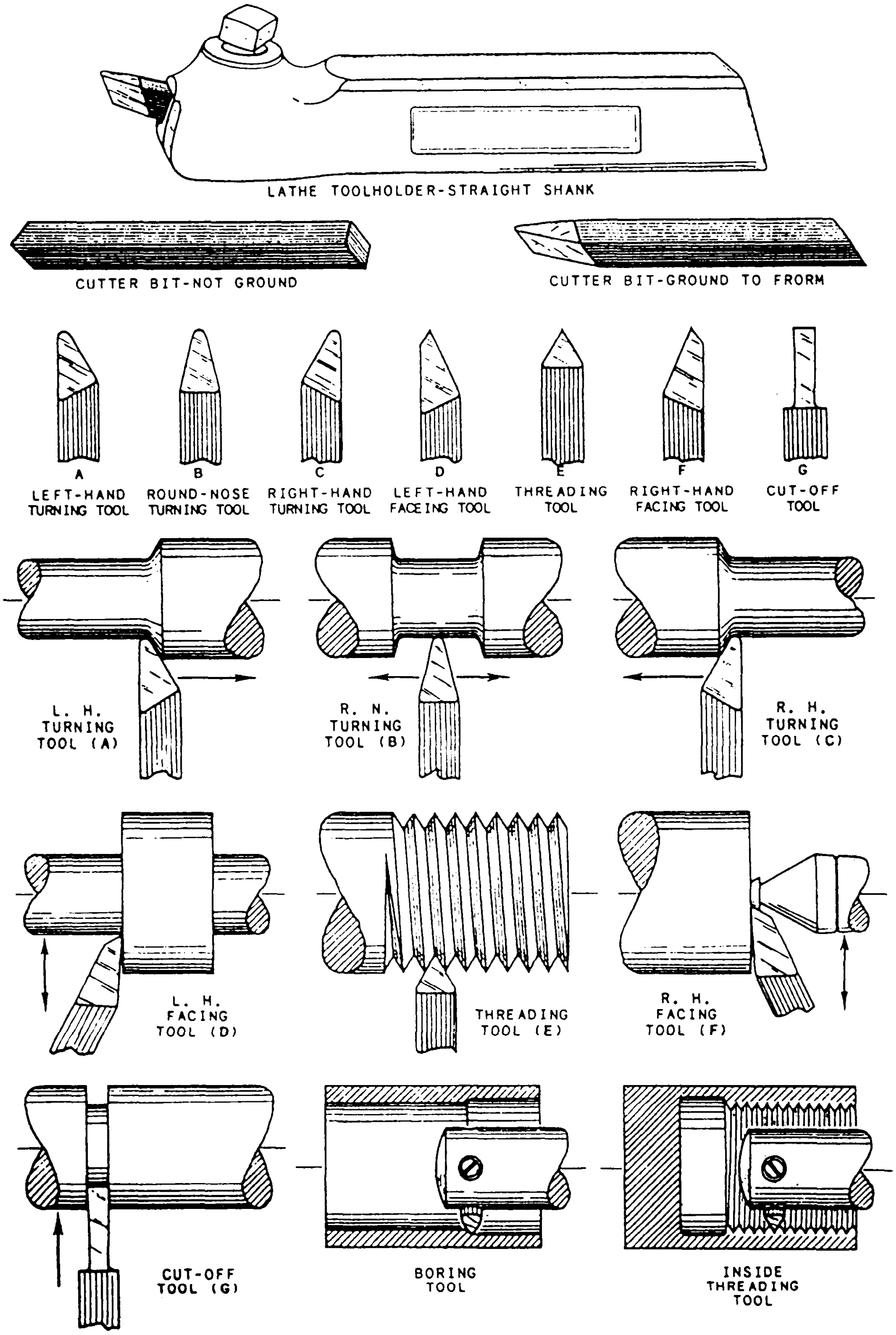


Figure 5-20.—Lathe tools and their application.

• **Internal-threading tool:** The internal-threading (inside-threading) tool is the same as the threading tool in figure 5-20, view E, except that it is usually much smaller. Boring and internal-threading tools may require larger relief angles when you use them in small diameter holes.

GRINDING HIGH-SPEED ENGINE LATHE CUTTING TOOLS

Machining techniques and the materials being machined limit the angles of a tool bit. However, when grinding the angles, you also must consider the type of toolholder and the position of the tool with respect to the axis of the workpiece. The angular offset and the angular vertical rise of the tool seat in a standard lathe toolholder affect the cutting edge angle and the end clearance angle of a tool when it is set up for machining. The position of the point of the tool bit with respect to the axis of the workpiece, whether higher, lower, or on center, changes the amount of front clearance.

Figure 5-21 shows some of the standard toolholders used in lathe work. Notice the angles at which the tool bits sit in the various holders. You must consider these angles with respect to the angles ground in the tools and the angle that you set the toolholder with respect to the axis of the work. Also, notice that a right-hand toolholder is offset to the **LEFT** and a left-hand toolholder is offset to the **RIGHT.** For most machining operations, a right-hand toolholder uses a left-hand turning tool and a left-hand toolholder uses a right-hand turning tool. Study figures 5-20 and 5-21 carefully to help you understand this apparent contradiction. Also, take into consideration that if you use a quick change toolpost and toolholder (fig. 5-22), your end relief angle and your back rake angle will change.

The contour of a cutting tool is formed by the side cutting edge angle and the end cutting edge angle of the tool. Views A through G of fig. 5-20 show the

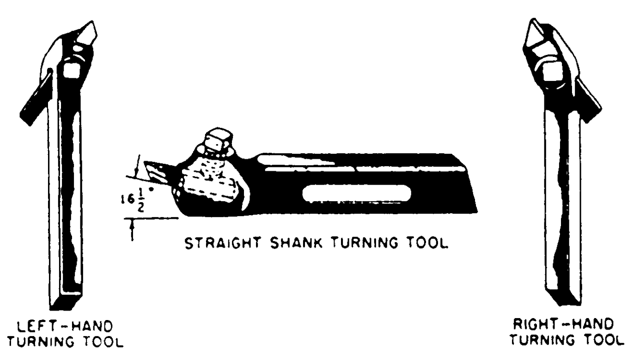
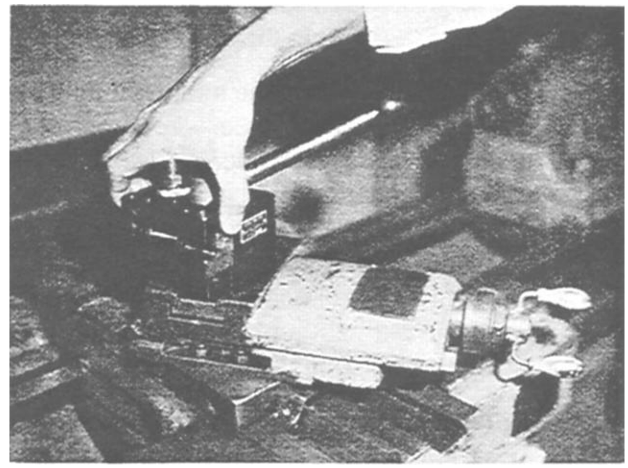


Figure 5-21.—Standard lathe toolholders.



28.302

Figure 5-22.—Quick change toolpost and toolholder.

recommended contours of several types of tools. There are no definite guidelines on either the form or the included angle of the contour of pointed tool bits, so you normally will forms the contour as you prefer. For roughing cuts, the included angle of the contour of pointed bits generally should be made as large as possible and still provide clearance on the trailing side or end edge. Tools for threading, facing between centers, and parting have specific shapes because of the form of the machined cut or the setup used.

The basic steps are similar when you grind a single-edged tool bit for any machine. The difference is in shapes and angles. *Machinery's Handbook* shows the recommended angles under the section on single-point cutting tools. Use a coolant when you grind tool bits. Finish the cutting edge by honing it on an oilstone. Figure 5-23 shows the basic steps you

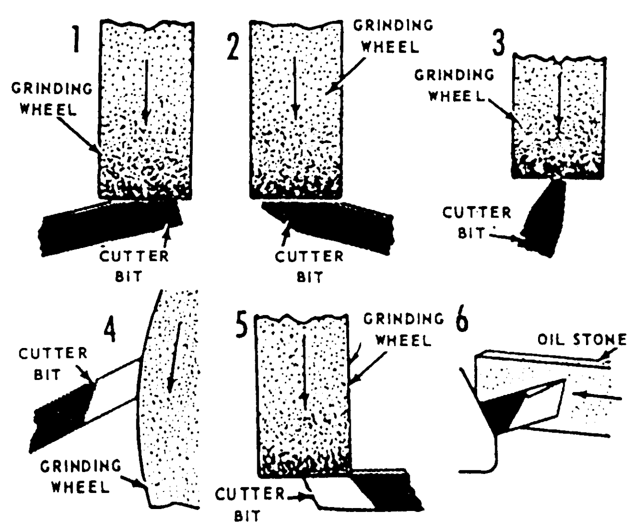


Figure 5-23.—Grinding and honing a lathe cutter bit.

should use to grind a round-nose turning tool. We'll describe each of the steps in the following paragraphs:

1. Grind the left side of the tool, holding it at the correct angle against the wheel to form the necessary side clearance. Use the coarse grinding wheel to remove most of the metal, then finish on the fine grinding wheel. (If you grind the cutting edge on the periphery of a wheel less than 6 inches in diameter, it will be undercut and will not have the correct angle.) Keep the tool cool while you grind it.

2. Grind the right side of the tool, holding it at the required angle to form the right side.

3. Grind the radius on the end of the tool. A small radius (approximately 1/32 inch) is preferable because a large radius may cause chatter. Hold the tool lightly against the wheel and turn it from side to side to produce the desired radius.

4. Grind the front of the tool to the desired front clearance angle.

5. Grind the top of the tool, holding it at the required angle to obtain the necessary side rake and back rake. Try not to remove too much of the metal. The more metal you leave on the tool, the better the tool will absorb the heat produced during cutting.

6. Hone the cutting edge all around and on top with an oilstone until you have a keen cutting edge. Use a few drops of oil on the oilstone. Honing will improve the cutting quality of the tool, produce a better finish on the work, and cause the cutting edge to stand up much longer than one that is not honed. The cutting edge should be sharp in order to shear off the metal rather than tear it off.

SHAPER AND PLANER TOOLS

Shaper and planer cutting tools arc similar in shape to lathe tools but differ mainly in their relief angles. These tools are held practically square with the work and do not feed during the cut; therefore, relief angles are much less than those in turning operations. The nomenclature of shaper and planer tools is the same as that for lathe tools; and the elements of the tool, such as relief and rake angles, are in the same relative positions as those shown in figure 5-18.

Several types of tools are required for shaper or planer operations. Although the types differ considerably in shape, the same general rules govern the grinding of each type.

To be sure you have an efficient cutting tool, grind the side relief and end relief of the tool to give a projecting cutting edge. If the clearance is insufficient, the tool bit will rub the work, causing excessive heat and producing a rough surface on the work. If the tool is given too much relief, the cutting edge will be weak and will tend to break during the cut. The fRont and side clearance angles should seldom exceed 3° to 5°.

In addition to relief angles, the tool bit must slope away from the cutting edge. This slope is known as side rake and reduces the power required to force the cutting edge into the work. The side rake angle is usually 10° or more, depending upon the type of tool and the metal being machined. Roughing tools should have no back rake although a small amount is generally required for finishing.

The shape and use of various standard shaper and planer cutting tools are illustrated in figure 5-24 and described in the following paragraphs:

- Roughing tool: This tool (fig. 5-24, view A) is very efficient for general use and is designed to take heavy cuts in cast iron or steel. You will generally grind it for left-hand operation as illustrated. For special applications, you can reverse the angles for right-hand cuts. Do not give this tool any back rake although the side rake may be as much as 20° for soft metals. Do finishing operations on small flat pieces with the roughing tool if a fine feed is used.

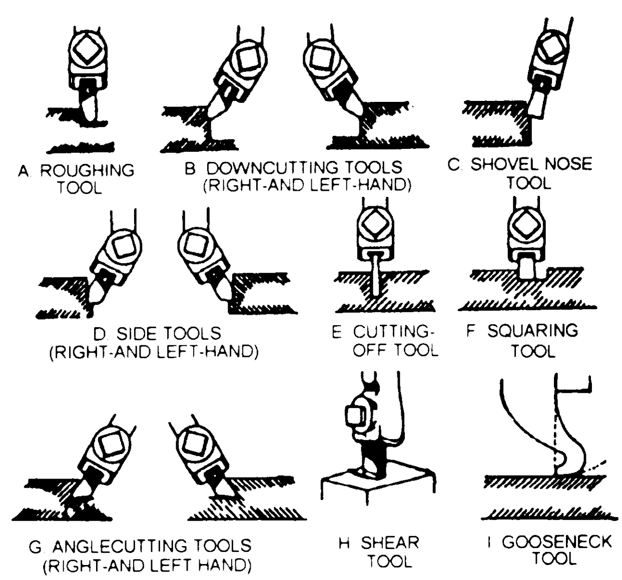


Figure 5-24.—Standard shaper and planer tools.

• **Downcutting tool:** You may grind and set this tool (fig. 5-24, view B) for either right- or left-hand operation and use it to make vertical cuts on edges, sides, and ends. It is substantially the same as the roughing tool, with the exception of its position in the toolholder.

• **Shovel-nose tool:** You may use this tool (fig. 5-24, view C) for downcutting in either a right- or left-hand direction. It requires a small amount of back rake and the cutting edge should be the widest part of the tool. Make the corners slightly rounded to give them longer life.

• **Side tool:** This tool (fig. 5-24, view D) comes in both right- and left-hand versions required to finish vertical cuts. You also may use these tools to cut or finish small horizontal shoulders to avoid changing tools after you make a vertical cut.

• **Cutting-off tool:** You should give this tool (fig. 5-24, view E) relief on both sides to allow free cutting action as the depth of cut is increased.

• **Squaring tool:** This tool (fig. 5-24, view F) is similar to a cutting-off tool and you can make it in any desired width. Use the squaring tool mostly to finish the bottoms and sides of shoulder cuts, keyways, and grooves.

• **Angle cutting tool:** This tool (fig. 5-24, view G) is adapted for finishing operations and is generally used following a roughing operation made with the downcutting tool. You may grind this tool for right- or left-hand operation.

• **Shear tool:** This tool (fig. 5-24, view H) is used to produce a high finish on steel and should be operated with a fine feed. Grind the cutting edge to form a radius of 3 to 4 inches, twisted to a 20° to 30° angle, and give it a back rake in the form of a small radius.

• **Gooseneck tool:** This tool (fig. 5-24, view I) is used to finish cast iron. You must forge it so the cutting edge is behind the backside of the tool shank. This feature allows the tool to spring away from the work slightly, reducing the tendency to gouge or chatter. Round off the cutting edge at the corners and give it a small amount of back rake.

GRINDING HANDTOOLS

You should keep hand tools in the best usable condition. To do that, you must sharpen cutting edges frequently and true or shape certain other tools for special purposes. Shape or sharpen chisels, punches, screwdrivers, and other handtools on an abrasive grinding wheel. We will explain the sharpening of these tools in the following paragraphs:

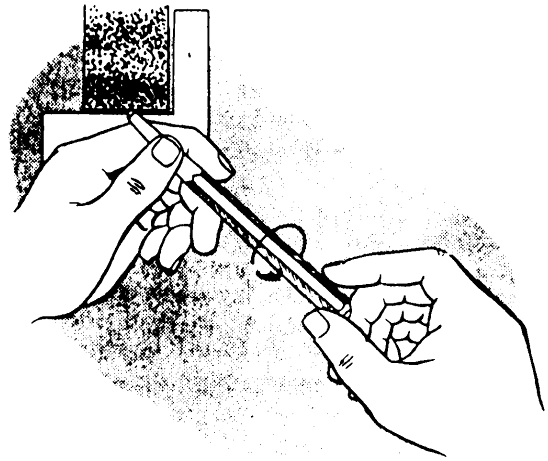


Figure 5-25.—Grinding a center punch with a bench grinder.

• **Center punches:** To sharpen a center punch, rest your hand on the tool rest of the grinder and cradle the end of the punch between the index finger and thumb of one hand, as shown in figure 5-25. Move the punch into light contact with the rotating grinder wheel, with the center line of the punch forming about a 45° angle with the face of the wheel. This will give the approximate 90° included angle required for a center punch. With the thumb and index finger of the other hand, rotate the punch as shown by the directional arrow in figure 5-25. Dip the punch in coolant frequently during the process.

• **Screwdriver tips:** Figure 5-26, views A and C, are the front views of a properly dressed common

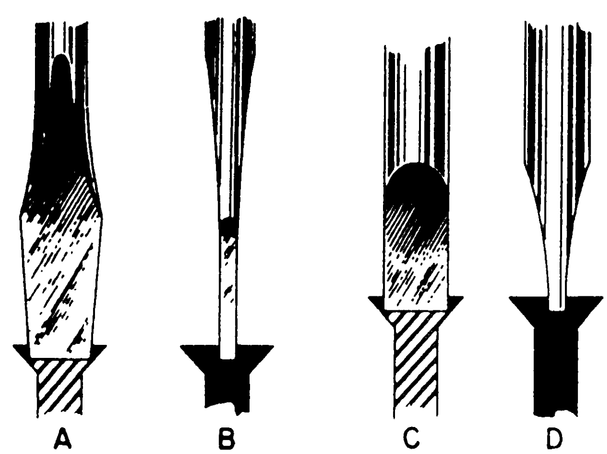


Figure 5-26.—Shapes of screwdrivers when properly dressed.

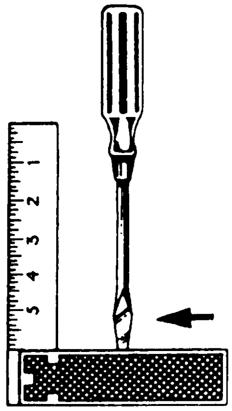


Figure 5-27.—Checking the squareness of the end of a screwdriver.

screwdriver. Views B and D are the side views. Dress the edges so the blade is symmetrical in shape, then square off the end. Check the squareness of the end by using a square. If the blade and shank appear to be parallel, the tip is square. See figure 5-27.

Next, grind the faces of the blade so they are parallel at the tip as shown in views B and D of figure 5-26. The thickness of the blade at the tip should be such that the tip will just enter the slot of the screw you intend to turn. With such a tip thickness, and the sides parallel, the screwdriver will have the least tendency to climb out of the screw slot when the screw is being turned. When grinding, do not let the tip get too hot or it will be softened.

- **Metal-cutting chisels:** These chisels are designed to cut cold metal, so we often use the general term *cold chisel* to describe them. The 60° angle shown in figure 5-28, is for a general-use cold chisel. Increase this angle to cut harder materials and decrease it for softer materials.

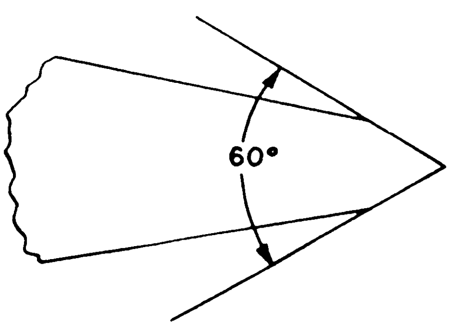


Figure 5-28.—Proper angle for a general use cold chisel.

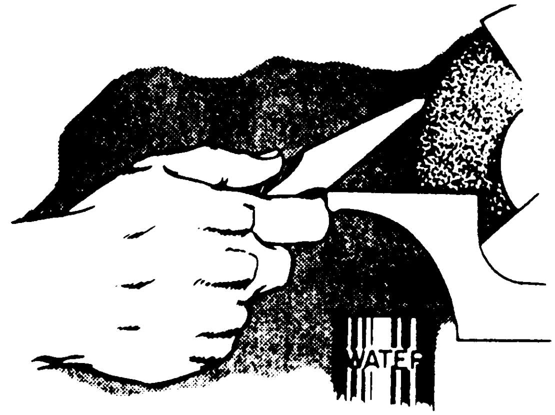


Figure 5-29.—Sharpening a cold chisel.

To sharpen this chisel, hold it to the wheel, resting it on the tool rest as shown in figure 5-29. Notice that the index finger, curved beneath the chisel, rides against the front edge of the tool rest. This ensures control of the chisel and will help you grind a single, equal bevel on each side.

Let the chisel rest lightly against the wheel while grinding. This will develop less heat and the air currents created by the wheel will have the maximum cooling effect. Be sure the cutting edge is kept cool or

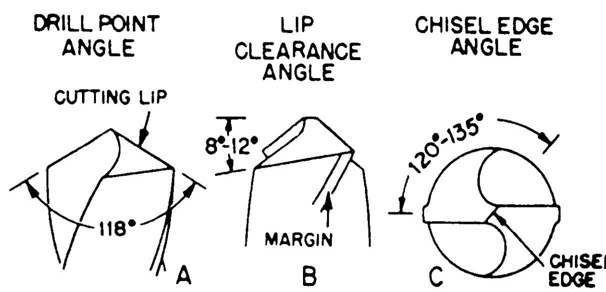


Figure 5-30.—Specifications for grinding a regular point twist drill.

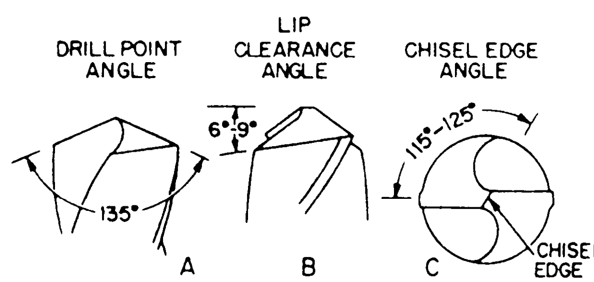


Figure 5-31.—Specifications for grinding a flat point twist drill.

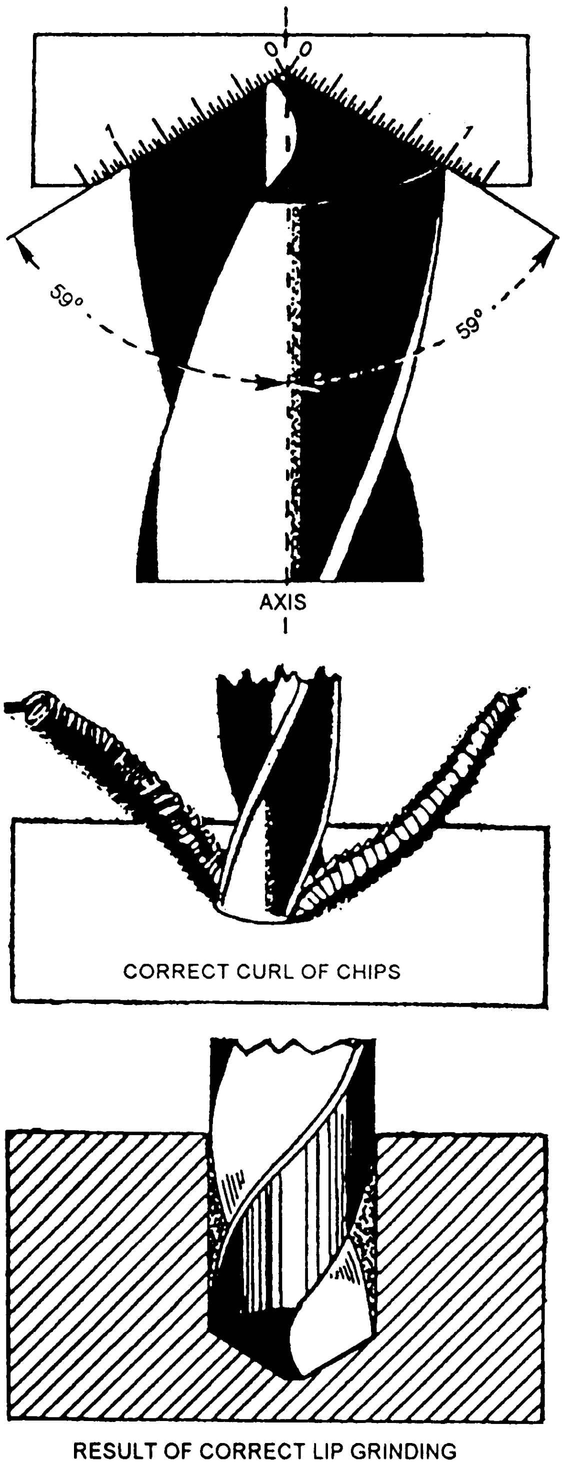


Figure 5-32.—Grinding drill lip correctly.

it may be softened. You may also want to dip the chisel in a coolant.

SHARPENING TWIST DRILLS

When grinding twist drills, it is most important that you meet the following criteria: (1) drill point angles must be equal and correctly sized, (2) cutting lips must be of equal length, (3) the clearance behind the cutting lips must be correct, and (4) the chisel-edge angle must be correct. All four are equally important when grinding either a regular point (fig. 5-30) used for general purposes, or a flat point (fig. 5-31) used to drill hard and tough materials.

Figure 5-32 shows the results of correct lip grinding and how equal drill point angles and two equal length cutting lips help achieve correct drill results.

Figure 5-33 shows a drill being checked during grinding. The drill-point gauge is being held against the body of the drill and has been brought down to where the graduated edge of the gauge is in contact with one cutting edge. In this way, both the drill-point angle and the length of the cutting edge (or lip) are checked at the same time. The process is repeated for the other side of the drill.

You determine lip clearance behind the cutting lip at the margin by inspection. This means you look at the drill point and approximate the lip-clearance angle (see figs. 5-30, view B, and 5-31, view B), or compare it to the same angle that has been set on a protractor. The lip-clearance angle is not necessarily a definite angle, but it must be within certain limits. Notice that this angle ranges from 8° to 12° in figure 5-30, view B and from 6° to 9° in figure 5-31, view B. Whatever angle in the range is used, however, lip clearance should be the same for both cutting lips.

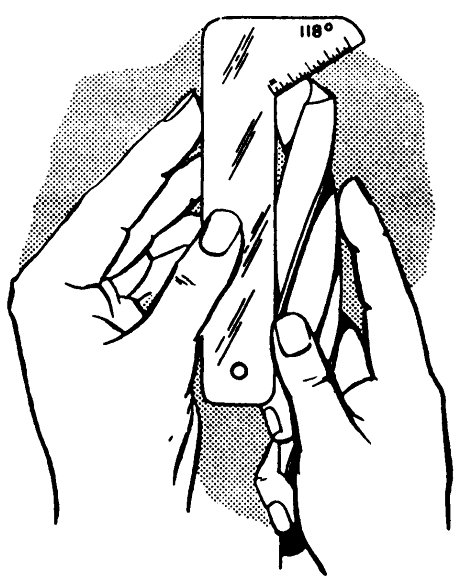


Figure 5-33.—Checking the drill point angle cutting edge.

There must be lip clearance behind the entire length of the cutting edge lip that extends from the margin of the drill to the chisel edge. This means that there must be "relief' behind the cutting lip along its entire length.

When you grind lip clearance, use the lip-clearance angle and the chisel-edge angle (shown in figs. 5-30, view C and 5-31, view C) as your guide to the amount of clearance you have ground into the drill behind the cutting lip along its entire length. The greater these angles are, the more clearance there will be behind their respective ends of the cutting lip. Too much lip clearance occurs when both the lip-clearance angle and the chisel-edge angle exceed their top limits. This weakens the cutting edge or lip by removing too much metal directly behind it. Too little or no lip clearance prevents the cutting edge from producing a chip, and the drill bit will not drill a hole.

To sharpen a twist drill, first ensure the grinder is ready. If necessary, dress the face of the wheel and adjust the toolrest. Start the grinder, let it come up to speed, and begin. Hold the twist drill as shown in figure 5-34, view A, which is a top view of the first step in grinding a drill. In the first step, be sure the axis of the drill makes an angle of about 59° (half of the drill-point angle) with the face of the wheel as shown in fig. 5-34, view A. Hold the cutting lip horizontal. Figure 5-35 is a side view of the same drill position shown in figure 5-34, view A.

The actual grinding of the drill point consists of three definite motions of the shank of the drill while you hold the point lightly against the rotating wheel. These three motions are (1) to the left, (2) clockwise rotation, and (3) downward.

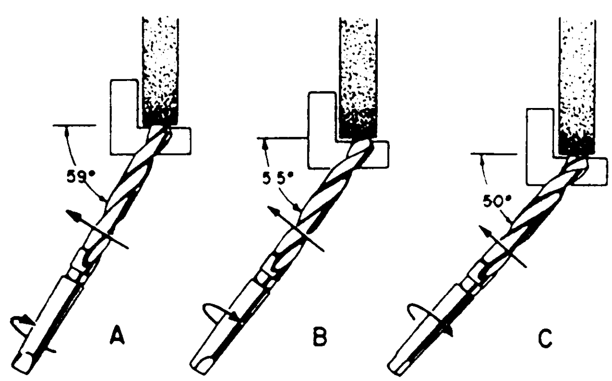


Figure 5-34.—Three steps for grinding a twist drill with a grinder.

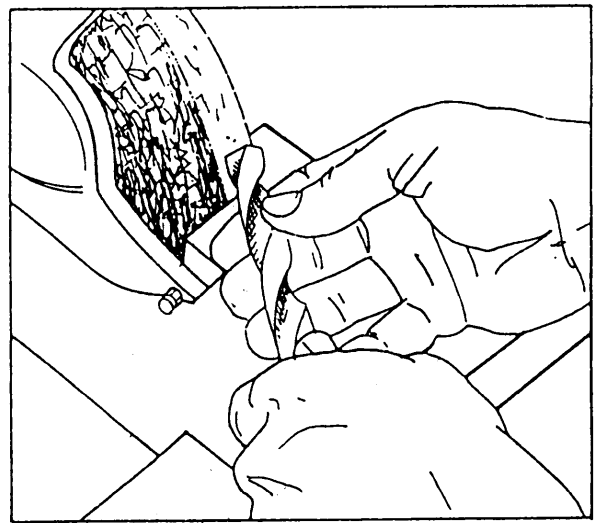


Figure 5-35.—Grinding a twist drill with a grinder (initial position.

Figure 5-34 shows the motion to the left in three views as the angle between the face of the wheel and the drill decreases from about 59° to about 50°.

In figure 5-34, the rotation arrows in views A, B, and C show the clockwise motion. The change in the position of the cutting lip and tang also shows rotation.

Because figure 5-34 is a top view, the downward motion is not noticeable. However all three motions are apparent when you compare the final position of the drill in figure 5-36 to the view in figure 5-35. All three motions taking place at the same time combine to produce the requirements mentioned earlier in this section: (1) equal and correctly sized drill-point angles, (2) equal-length cutting lengths, (3) correct clearance behind the cutting lips, and (4) correct

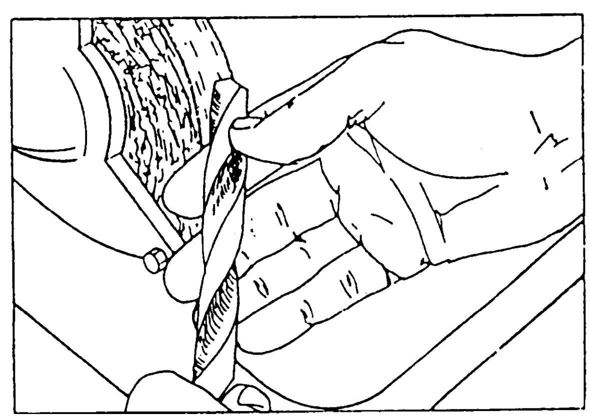


Figure 5-36.—Grinding a twist drill with a grinder (final position.

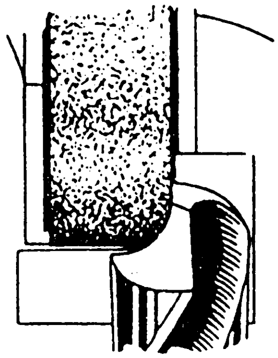


Figure 5-37.—Grinding a twist drill for brass.

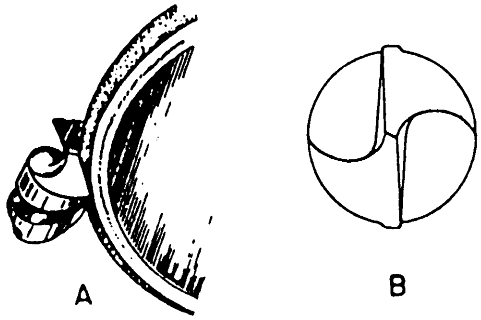


Figure 5-38.—Thinning the web of a twist drill.

chisel-edge angle. Use a drill-point gauge (fig. 5-33) and inspection to be sure you have met these four requirements.

SHARPENING A TWIST DRILL TO DRILL BRASS

To sharpen a drill to drill brass, hold the cutting lip against the right side of the wheel as shown in figure 5-37. Grind the flute slightly flat, in line with the axis of the drill, to greatly reduce the included angle of the cutting lip. This will give the drill the scraping action needed for brass rather than the cutting action used for steel. It will prevent the tendency of the drill bit to be sucked into the hole being drilled. This can be especially troublesome when you drill through a pilot hole.

THINNING THE WEB OF A TWIST DRILL

Repeated sharpening shortens a drill and that increases the web thickness at the point. This may require web thinning.

To thin the web of a drill, hold the drill lightly to the face of a round-faced wheel, as shown in figure 5-38, view A, and thin the web for a short distance behind the cutting lip and into the flutes. This is shown in figure 5-38, view B. Notice that the cutting lip is actually (but only slightly) ground back, reducing its included angle a small amount but not enough to affect the operation of the drill.